



Tekla Structures

Analysis Manual

Product version 10.0
November 2003

Copyright © 2003 Tekla Corporation

Copyright© 1992-2003 Tekla Corporation. All rights reserved.

This Software Manual has been developed for use with the referenced Software. Use of the Software, and use of this Software Manual are governed by a License Agreement. Among other provisions, the License Agreement sets certain warranties for the Software and this Manual, disclaims other warranties, limits recoverable damages, defines permitted uses of the Software, and determines whether you are an authorized user of the Software. Please refer to the License Agreement for important obligations and applicable limitations and restrictions on your rights.

In addition, this Software Manual is protected by copyright law and by international treaties. Unauthorized reproduction, display, modification, or distribution of this Manual, or any portion of it, may result in severe civil and criminal penalties, and will be prosecuted to the full extent permitted by law.

Tekla, Tekla Structures, Xcity, Xengineer, Xpipe, Xpower, Xsteel, and Xstreet are either registered trademarks or trademarks of Tekla Corporation in the European Union, the United States, and/or other countries. Other product and company names mentioned in this Manual are or may be trademarks of their respective owners. By referring to a third-party product or brand, Tekla does not intend to suggest an affiliation with or endorsement by such third party and disclaims any such affiliation or endorsement, except where otherwise expressly stated.

Elements of the software described in this Manual may be the subject of pending patent applications in the European Union and/or other countries.

Contents

Preface	1
 1 Getting Started with Analysis	5
1.1 Basics	6
1.2 Determining member properties	8
1.3 Analysis information and settings	18
 2 Loads	25
2.1 Basics	25
2.2 Grouping loads.....	27
2.3 Load types and properties	30
2.4 Distributing loads.....	35
2.5 Working with loads	37
2.6 Load reference	40
 3 Analysis and Design	41
3.1 Analysis model properties	42
3.2 Load combination	49
3.3 Working with analysis and design models	56
3.4 Structural design	60
3.5 Analysis and design reference	61
 Index	63

Preface

Introduction

This is the **Tekla Structures Analysis Manual**, a comprehensive guide to Tekla Structures analysis and design features. The following paragraphs explain how this guide is organized, suggest different paths for different types of user, describe the other guides provided in the package, and tell you how to report any problems you have with the software or guides.

Audience

This guide is aimed at structural engineers who analyze and design concrete and steel structures.

We assume that you are familiar with the processes of structural engineering.

Conventions used in this guide

Typefaces

We use different typefaces for different items in this guide. In most cases the meaning is obvious from the context. If you are not sure what a certain typeface represents, you can check it here.

- Any text that you see in the user interface appears in **bold**. Items such as window and dialog box titles, field and button names, combo box options, and list box items are displayed in this typeface.
- New terms are in ***italic bold*** when they appear in the current context for the first time.
- All the text you enter yourself appears in "quotation marks".
- We use *italics* for emphasis.
- Extracts of Tekla Structures's program code, HTML, or other material that you would normally edit in a text editor, appears in mono-spaced Courier font.

- Program names, such as functions, environment variables, and parameters, appear in **Courier bold**.
- Filenames and folder paths appear in Arial.

Noteboxes

We use several types of noteboxes, marked by different icons. Their functions are shown below:



A **tip** might introduce a shortcut, or suggest alternative ways of doing things. A tip never contains information that is absolutely necessary.



A **note** draws attention to details that you might easily overlook. It can also point you to other information in this guide that you might find useful.



You should always read very **important notes and warnings**, like this one. They will help you avoid making serious mistakes, or wasting your time.



This symbol indicates **advanced or highly technical information** that is usually of interest only to advanced or technically-oriented readers. You are never required to understand this kind of information.

Related guides

Tekla Structures includes a comprehensive help system in a series of online books. You will also receive a printed installation guide with your setup CD.

- **Modeling Manual**
How to create a physical model.
- **Analysis Manual**
How to create loads and run structural analysis.
- **Detailing Manual**
How to create reinforcement, connections, and details.
- **Drawing Manual**
How to create and edit drawings.
- **System Manual**
Covers advanced features and how to maintain the Tekla Structures environment.
- **TplEd User's Guide**
How to create and edit report and drawing templates.
- **SymEd User's Guide**
How to use the SymEd graphical interface to manipulate symbols.
- **Installation Guide**
Printed booklet explaining how to install Tekla Structures.

Organization

This guide is divided into the following chapters and appendices:

Chapter 1: Getting Started with Analysis

Explains how to prepare your Tekla Structures model for structural analysis and design.

Chapter 2: Loads

Describes how to create, manage, and group loads in Tekla Structures.

Chapter 3: Analysis and Design

Explains how to run structural analysis in Tekla Structures.

Chapter 4: Analysis Engine

Explains how to use *STAAD.Pro* as analysis engine in Tekla Structures.

1

Getting Started with Analysis

Introduction

This chapter explains how to prepare a Tekla Structures model for structural analysis and design. It includes a general description of the principles of analysis and design and discusses the theoretical basis of the analysis method used in Tekla Structures. This chapter also explains what is included in the analysis model, and how it is included. You will also learn how to define support conditions for parts.

Audience

This chapter is for engineers and designers who run structural analysis on concrete and steel structures.

Assumed background

We assume that you have read [Parts](#) in the Modeling Manual and created parts.

Contents

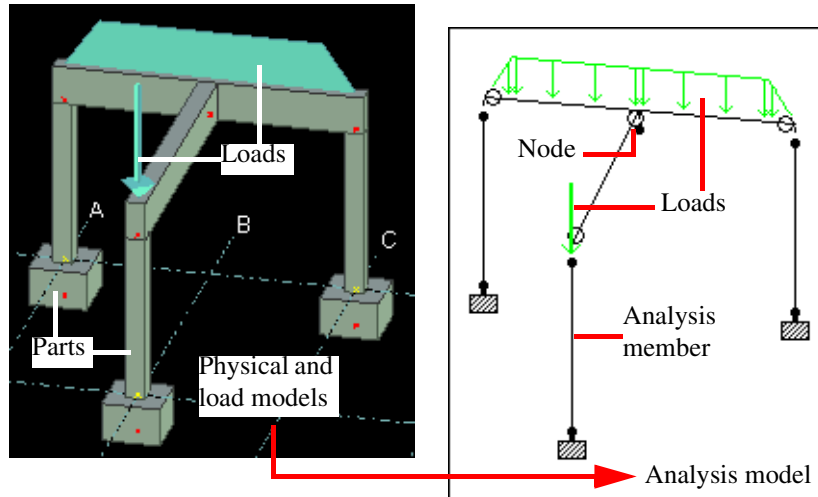
This chapter is divided into the following sections:

- [Basics](#) (p. 6)
- [Determining member properties](#) (p. 8)
- [Analysis information and settings](#) (p. 18)

1.1 Basics

In this section

This section presents the basic vocabulary and concepts we use to describe structural analysis in Tekla Structures. For more definitions of terms, see the [Glossary](#) in the Modeling Manual. The illustrations below show the analysis concepts and procedures.



Physical model

A **physical model** includes the parts you have created using the Model Editor, and information related to them. Each part in the physical model exists in the completed structure.

Load model

The **load model** contains information about loads and load groups. It also contains information about the building code Tekla Structures uses in load combination. To create a load model, see [Loads](#) (p. 25).

Analysis model

Tekla Structures generates an **analysis model** of the physical and load models when you run structural analysis. Tekla Structures does the following in order to generate the analysis model:

- Creates nodes and analysis members and elements of the physical parts
- Determines the support conditions for nodes
- Determines the connectivity between the members and nodes
- Distributes loads to members and elements

The analysis model also includes load combinations.

Analysis engine

Tekla Structures uses *STAAD.Pro* as its **analysis engine**. The engine processes and calculates analysis models using the finite element method (FEM). It uses data from the analysis model to generate analysis results.



Tekla Structures analyzes parts using properties in the profile and material catalogs, including user-defined properties. If there are no profile or analysis properties in the catalog, Tekla Structures calculates them using the profile dimensions in the model.

Following through structural analysis

Before analysis

Carry out the following steps before you run structural analysis in Tekla Structures:

1. Create the main load-bearing parts to form the physical model. See [Parts](#) in the Modeling Manual. There is no need to detail or create connections at this stage.



To create accurate analysis models, make sure that connected parts have common reference points, for example, at grid line intersections.

2. Set the support conditions for parts and connections, as well as other analysis properties for individual members. See [Determining member properties](#) (p. 8).
3. Create the load model. See [Loads](#) (p. 25).
4. Create a new analysis model and define its properties. See [Analysis model properties](#) (p. 42).
5. Create load combinations.

Now you are ready to run the analysis.

See also

The following sections discuss the theoretical basis of the analysis method used in Tekla Structures. They also explain what is included in the analysis model, and how it is included.

- [Members, elements, and nodes](#) (p. 8)
- [A closer look at the analysis model](#) (p. 19)
- [Loads in analysis](#) (p. 21)
- [Load modeling code](#) (p. 22)
- [Analysis method](#) (p. 23)

Members, elements, and nodes

Members	Every physical part (beam or column) that you select to include in the analysis model produces one or more analysis members . A single physical part produces several members if the part intersects with other parts. Tekla Structures splits the part at the intersection points of the member axes.
Elements	Tekla Structures splits the plates, slabs, and panels that you include in the analysis model into analysis elements , based on their analysis properties and the parts connected to them.
Nodes	<p>Nodes connect analysis members and elements. Tekla Structures creates nodes at:</p> <ul style="list-style-type: none">• The ends of members• The intersection points of member axes• The corners of elements

The following properties affect the exact location of nodes:

- Part profiles, i.e. neutral axis and orientation
- Part reference lines (see [Part location](#) in the Modeling Manual)
- Location of member axes (see [Member axis location](#) (p. 10) and [Member axis](#) (p. 44))
- Location and shape of elements (see [Analysis properties of plates](#) (p. 11))
- [Node definition method](#) (p. 45)

To force members to meet in the analysis model, Tekla Structures may need to merge nodes, shift or extend member axes, create rigid links between nodes, ignore minor members, etc.

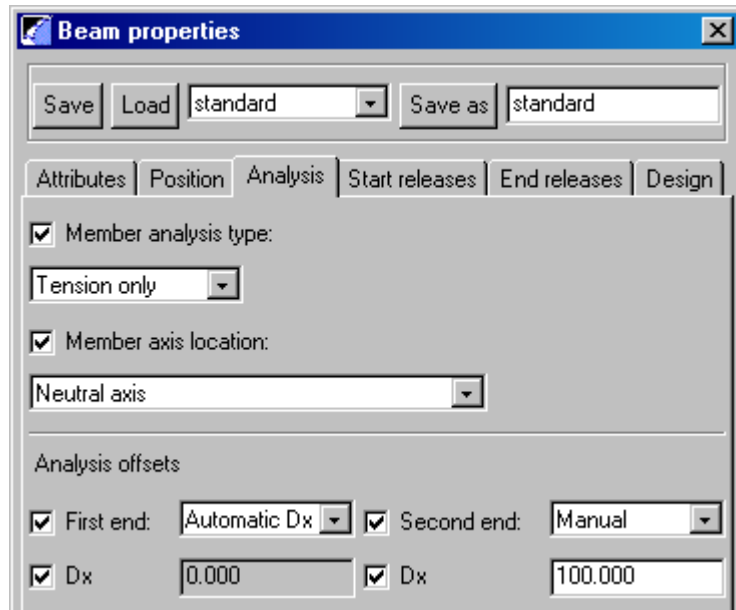


The methods used to create a physical model affect the analysis model. Because of this, you may need to try different modeling methods and analysis model properties in order to create an accurate analysis model of a complex physical model.

For more information on where and how Tekla Structures creates nodes, members, and elements, see [A closer look at the analysis model](#) (p. 19).

1.2 Determining member properties

You can define analysis properties for individual parts, or for an entire analysis model. This section describes the properties of the individual analysis members. To define these, use the **Analysis**, **Start releases**, **End releases**, and **Design** tabs in the part properties dialog boxes, or the **Analysis** tab in the connection and detail dialog boxes.



For more information on using common properties for the parts in an analysis model, see [Analysis model properties](#) (p. 42).

Member analysis type

Use the **Analysis** tab in the part properties dialog boxes to define how Tekla Structures handles individual members in the analysis. The following table lists the options.

You can have Tekla Structures show the member analysis type of parts using different colors in the physical model. Click **Setup > Colors...** and select **Analysis type** in the **Color by** list box. See also [General settings](#) in the Modeling Manual.

Option	Description	Color
Normal	Member may take any type of load.	White
Ignore	Member ignored in the analysis.	Red
Truss	Member can only take axial forces, not bending or torsion moments, or shear forces.	Green
Truss - Tension only	Member can only take tensile axial forces, not moments or shear forces. If this member goes into compression, it is ignored in the analysis.	Cyan

Option	Description	Color
Truss - Compression only	Member can only take compressive axial forces, not moments or shear forces. If this member goes into tension, it is ignored in the analysis.	Yellow
Rigid diaphragm	Only applies to contour plates, concrete slabs, and concrete panels parallel to the global xy, yz, or zx plane. All nodes of this member are connected with rigid links and their displacements affect each other.	Blue

For more information on members with the **Truss**, **Tension only**, or **Compression only** setting, see [A closer look at the analysis model \(p. 19\)](#).

Member axis location

The locations of the member axes of parts define where the analysis members actually meet, and their length in the analysis model. They also affect where Tekla Structures creates nodes. See [Members, elements, and nodes \(p. 8\)](#) and [A closer look at the analysis model \(p. 19\)](#).

Use the **Analysis** tab in the part properties dialog boxes to define the member axis location of individual parts for analysis purposes. The options are:

Option	Description
Neutral axis	The neutral axis is the member axis for this part. The location of the member axis changes if the profile of the part changes.
Reference axis	The part reference line is the member axis for this part. See also Part location in the Modeling Manual.
Reference axis (eccentricity by neutral axis)	The part reference line is the member axis for this part. The location of the neutral axis defines axis eccentricity.

Tekla Structures uses the options above for each part when you select the **Model default** option for the member axis location in the analysis model properties. See [Member axis \(p. 44\)](#).



If you select the **Neutral axis** option, Tekla Structures takes the part location and end offsets into account when it creates nodes. See **End offsets** in the Modeling Manual. If you select either of the **Reference axis** options, Tekla Structures creates nodes at part reference points.

Analysis member offsets

Use offsets at the ends of analysis members to shorten or lengthen members in their local x directions, for analysis purposes and to take the eccentricity effects into account.

For example, if a beam only actually spans the clear distance between two supporting columns, you can use offsets to only include the clear distance in the analysis, instead of the distance between the center points of the columns.

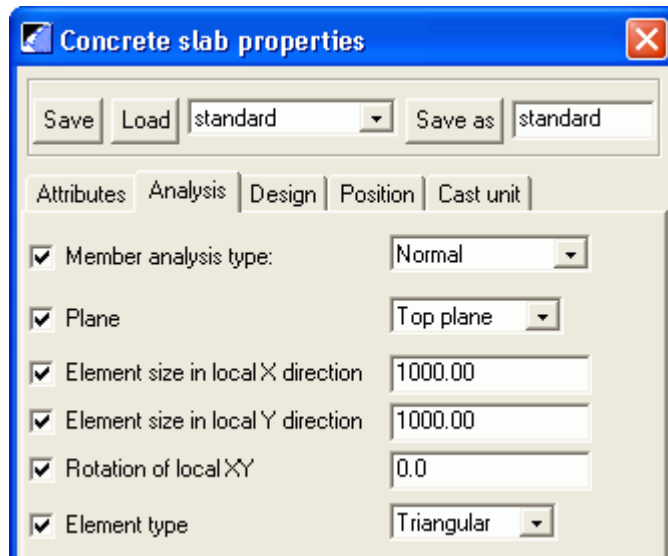
Use the **Analysis** tab in the part properties or connection dialog boxes to define the offset at each end of a member. The options are:

Option	Description
Manual	Works like end offsets for parts in the physical model. Enter a positive or negative value in the Dx field. See also End offsets in the Modeling Manual.
Automatic Dx	The offset is the distance between the intersection of the parts' neutral axes and the intersection of the edges of the parts.
Longitudinal member offset	Only applies to connection members and details. Works like the Manual option and a value in the Dx field for parts.

Analysis properties of plates

When creating an analysis model, Tekla Structures creates analysis elements for contour plates, concrete slabs, and concrete panels.

Use the **Analysis** tab in the appropriate part properties dialog boxes to define how Tekla Structures creates analysis elements.



The analysis properties of plates are:

Property	Description
Member analysis type	See Member analysis type (p. 9) . Set to Normal to create elements in the analysis model.
Plane	The plane of the plate on which Tekla Structures creates the elements. The options are Top plane or Bottom plane . The reference points of connected parts must also be in this plane.
Element size	The approximate dimensions of the elements, in the local x and y directions of the plate. For triangular elements, the approximate dimensions of the bounding box around each element.
Rotation of local xy	The target rotation angle of the elements, in the local xy plane of the plate. The first two points you pick when you create a plate define the local x direction of the plate.
Element type	The shape of the elements. The options are Triangular or Quadrilateral .

Analysis properties of components

Use the **Analysis** tab in the connection or detail dialog boxes to define how Tekla Structures handles connections and details in the analysis.

Picture | Parts | Parameters | General | Notch | Bolts | Design | Analysis

Use analysis restraints ☒ Yes

Member selection ☒ Primary

Restrain combination ☒

Support condition ☒ Connected

Ux Uy Uz Rx Ry Rz

Ux	<input checked="" type="checkbox"/> Free	0.00
Uy	<input checked="" type="checkbox"/> Free	0.00
Uz	<input checked="" type="checkbox"/> Free	0.00
Rx	<input checked="" type="checkbox"/> Pinned	0.00
Ry	<input checked="" type="checkbox"/> Pinned	0.00
Rz	<input checked="" type="checkbox"/> Pinned	0.00

Longitudinal member offset ☒ 0.00

Analysis profile ☒ ...

Analysis profile length ☒ 0.00

The analysis properties of connections and details are:

Property	Description
Use analysis restraints	<p>Set to Yes to use the analysis properties of the connection or detail in the analysis, instead of the analysis properties of the parts in the connection.</p> <p>You must also select the By connection checkbox against Member end release method in the Analysis model attributes dialog box when you create the analysis model. See Member end connectivity (p. 44).</p>
Member selection	<p>Use to associate the analysis properties with each connection part (Primary, 1. secondary, 2. secondary, etc.).</p>

Property	Description
Restraint combination	See Support conditions (p. 14) and Defining support conditions (p. 14).
Support condition	
Longitudinal member offset	See Analysis member offsets (p. 11).
Analysis profile	Tekla Structures uses this profile in the analysis, instead of the one in the physical model, in order to take the stiffness of the connection or detail into account.
Analysis profile length	This means that in the analysis, Tekla Structures overrides the profile of the part in the physical model, for this length.

Support conditions

In structural analysis, the stresses and deflections of a part depend on how it is supported by, or connected to, other parts. You normally use restraints or springs to model connections. These determine how analysis members move, deflect, warp, deform, etc., in relation to each other, or to nodes.

Member ends and nodes have degrees of freedom (DOF) in three directions. The displacement of a member end can be free or fixed, and the rotation can be pinned or fixed. If the degree of connectivity is between free, or pinned, and fixed, use springs with different elastic constants to model them.

Tekla Structures uses part, connection, or detail properties to determine how to connect members in the analysis model. To define the member end conditions, use the **Start releases** and **End releases** tabs in the part properties dialog boxes. The connection and detail dialog boxes have **Analysis** tabs.

The analysis properties of a member determine the degrees of freedom for each end of a main part or member. The first end of a part has a yellow handle, the second end has a magenta handle. See also [Part location](#) in the Modeling Manual.

Defining support conditions

Parts






Use the **Start releases** and **End releases** tabs in the part properties dialog boxes to define support conditions. The **Start releases** tab relates to the first part end (yellow handle), the **End releases** tab to the second part end (magenta handle).

Connections and details

Support conditions

Use the **Analysis** tab in the connection or detail dialog boxes to define the support conditions for the members and node in a connection. Use the **Member selection** list box to associate the support conditions with each connection part (**Primary**, **1. secondary**, **2. secondary**, etc.).

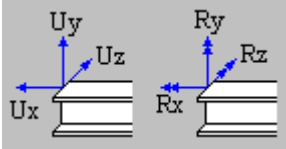
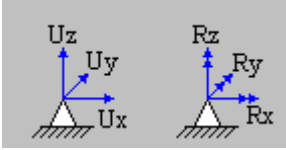
Tekla Structures includes four predefined combinations for member ends, and an option for user-defined settings. The predefined combinations (the first four in the following table) automatically set the appropriate support conditions and degrees of freedom. The combinations are:

Combination	Support condition	Translational DOFs	Rotational DOFs
	Supported	Fixed	Fixed
	Supported	Fixed	Pinned
	Connected	Fixed	Fixed
	Connected	Fixed	Pinned
	Use this option to define your own settings for the supports and connections at member ends. You can use springs and almost any combination of degrees of freedom.		



To ensure that the part remains stable, and that all loads applied to it pass through to the other structures, avoid using combinations with too many degrees of freedom.

The support conditions of a member end can be:

Option		Description
Connected		<p>Member end is connected to an intermediate analysis node (another part).</p> <p>Indicate degrees of freedom for the node.</p>
Supported		<p>Member end is the ultimate support for a superstructure (for example, the foot of a column in a frame).</p> <p>Indicate degrees of freedom for the support.</p>

Displacements and rotations

'U' denotes translational degrees of freedom (displacement). 'R' denotes rotational degrees of freedom (rotation). Define the degrees of freedom in the global coordinate system. The options are:

Option	More information
Free	Only applies to translational degrees of freedom.
Pinned	Only applies to rotational degrees of freedom.
Fixed	
Spring	Enter translational and rotational spring constants. The units Tekla Structures uses depend on the program's unit settings. See Units and decimals in the Modeling Manual.
Partial release	<p>Only applies to rotational degrees of freedom.</p> <p>Use to specify the degree of connectivity, if it is between fixed and pinned. Enter a value between 0 (fixed) and 1 (pinned).</p>

Design information

Use the **Design** tab in the part properties dialog boxes to view and modify the design properties of individual parts in an analysis model. Design properties are properties which can vary, according to the design code and the material of the main part (for example, design settings, factors, and limits).

Strip footing properties

Save Load standard Save as standard

Attributes Position Cast unit Bending Analysis Start releases End releases **Design**

Code **EC2(STAAD)** Analysis model **footing**

	Name	Use default	Value	Unit
1	Kmode - Buckling length by analysis member length	1	No	
1	Ky - Length factor for Buckling (Y)	0	0.00	
1	Kz - Length factor for Buckling (Z)	0	0.00	
1	Width - Width of cross section to be used in beam design	0	250.00	mm
1	Depth - Depth of cross section to be used in beam design	1	0.00	mm
1	Brace - Column bracing	1	0 = Col...	
1	Mmag - Factor by which column design moments to be ...	1	1.00	
1	Fc - Compressive strength of concrete	1	40.00	N/mm ²
1	Fymain - Yield strength for main reinforcement steel	1	400.00	N/mm ²
1	Fysecc - Yield strength for secondary steel	1	400.00	N/mm ²
1	Clear - Distance from surface of member to edge of outer...	1	25.00	mm
1	Minmain - Minimum main reinforcement bar size	1	0.00	mm
1	Minsec - Minimum secondary reinforcement bar size	1	0.00	mm
1	Maxmain - Maximum main reinforcement bar size	1	0.00	mm
1	Nsection - Number of equality spaced sections for beam ...	1	10	
1	Track - Track parameter	1	0 = Be...	
1	Serv - Serviceability checks	1	0 = No...	

OK Apply Modify Get ☒ / ☐ Cancel

The properties you see when you first open the dialog box are the properties that apply to the entire analysis model you have selected in the **Analysis & Design models** dialog box. See also [Design codes and methods](#) (p. 47).

To set different design properties for specific parts, modify the values in the appropriate part properties dialog box.

For example, if the analysis model contains parts with different material grades, define the most common material grade using the analysis model properties. Then change the material grade of specific parts using the appropriate part properties dialog box.

Properties of intermediate members

When creating an analysis model, Tekla Structures may need to produce more than one analysis member for each physical part. This can result in intermediate members and member ends.

Tekla Structures determines the analysis properties of intermediate members as follows:

1. The member analysis type and member axis location of the analysis members are the same as of the original part.
2. The analysis offsets of the part ends apply to the corresponding analysis member ends. Intermediate member ends do not have analysis offsets.
3. The support conditions of all intermediate member ends are **Connected**. The translational and rotational degrees of freedom are all **Fixed**. This reflects the nature of the physical part, which is a continuous length.
4. The effective buckling length of each analysis member is $K \cdot L$. K is the length factor for buckling. L is:
 - The analysis member length. To use this option, set the part's **Kmode** design property to **Yes**.
 - The length of the original part. To use this option, set the part's **Kmode** design property to **No**.
5. The other design properties are the same for the analysis members as for the original part.

1.3 Analysis information and settings

This section generally discusses the analysis process and describes analysis settings.

Analysis engine

Tekla Structures uses *STAAD.Pro* as its analysis engine.

First you create the physical, load, and analysis models using Tekla Structures. Then *STAAD.Pro* uses the information from these models to run the analysis. You view the analysis results using the *STAAD.Pro* postprocessor.



The Tekla Structures models contain all the input data for the analysis. You cannot change this data in *STAAD.Pro*.

For more information on *STAAD.Pro*, see [Introduction](#) in the online help.

A closer look at the analysis model

This section gives detailed information on how Tekla Structures creates analysis models of physical models.



The methods used to create a physical model affect the analysis model. Because of this, you may need to try different modeling methods and analysis model properties in order to create an accurate analysis model of a complex physical model.

Objects

Tekla Structures ignores the following objects in the analysis, even if you have included them in the analysis model (see [Objects in an analysis model \(p. 43\)](#)):

- Parts and loads that are filtered out (see [Analysis model filter \(p. 43\)](#))
- Connection objects (minor parts, bolts, reinforcing bars, etc.)
- Parts with the **Ignore** setting (see [Member analysis type \(p. 9\)](#))

Truss members

Tekla Structures does not split members with the **Truss**, **Tension only**, or **Compression only** setting (truss members). This is how Tekla Structures handles them:

Conditions	Action
Two or more truss members intersect.	Tekla Structures splits none of the members.
<ol style="list-style-type: none">1. A truss member intersects with a normal member.2. The intersection is not close to a node (outside the merge distance, see the table below).	Tekla Structures does not split either of the members.
<ol style="list-style-type: none">1. A truss member intersects with a normal member.2. The normal member end meets with the truss member axis.	<p>Tekla Structures changes the truss member to a normal member. Tekla Structures has to split the changed member to prevent the normal member end from losing support.</p> <p>Tekla Structures will ask you if you want to select and check the affected parts in the physical model.</p>

Nodes connecting members and elements

Tekla Structures first creates analysis nodes:

- On member axes at the ends of parts
- At the intersection points of member axes
- At the corners of elements

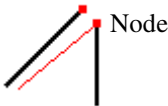
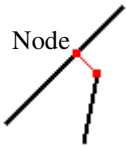
Tekla Structures then checks if the analysis members have common nodes. If they do not, Tekla Structures uses the methods described in the following table to connect members.

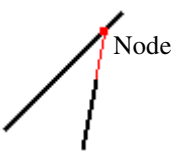
Common node

To have a common node in the analysis, physical parts must collide or be not further than 50 mm from each other's exact part solid.

Other methods

If physical parts (beams and columns) collide but their member axes do not intersect, Tekla Structures may need to carry out some of the following actions to make the members meet to create common nodes in the analysis model:

Action	Conditions	More information
Merge nodes and shift member axes	<p>Nodes are close to each other (within the merge distance).</p> <p>To define the merge distance, set the environment variable XS_AD_NODE_COLLISION_CHECK_DISTANCE. The default value is 10 mm.</p>	<p>Tekla Structures shifts the axis of the member which is closest to being horizontal, by moving its node. Tekla Structures does not shift the member axes of vertical parts (for example, columns).</p> 
Create rigid links between nodes	<ol style="list-style-type: none"> 1. Nodes are outside the merge distance. 2. You have selected the Use rigid links option in the analysis model properties. See Defining nodes (p. 45). 	

Action	Conditions	More information
Extend member axes	<ol style="list-style-type: none"> Nodes are outside the merge distance. You have selected the Force to centric connection option in the analysis model properties. See Defining nodes (p. 45). The resulting node is within 100 mm from the original nodes. 	
Ignore minor members	<ol style="list-style-type: none"> Members are shorter than 5 mm. Members do not have loads on them, and self-weight is not included in load combinations. Members are not supports. 	

Element nodes

This is how Tekla Structures creates nodes when plates connect with other parts:

Connected part	Action
Beam	Tekla Structures splits the beam and creates nodes in it at the element corners.
Column	Tekla Structures creates a node at the column.
Another plate	Tekla Structures creates the analysis elements so that the plates have common nodes on the edges of the plates.

See also

[Stiffness Analysis](#) in the online help

[Plate and Shell Element](#) in the online help

Loads in analysis

These are the principles that Tekla Structures follows when it processes loads in the physical model to create analysis model loads.

You define which loads are included in the analysis model. Tekla Structures applies these loads to members, based on the part name filtering criteria and each load's bounding box. See [Applying loads to parts \(p. 36\)](#).

Point loads	Point loads are transferred to the node that is generated from the member to which the load is applied to, and that is nearest to the location of the load, or to a member. Only one load can correspond to each physical load.
Line loads	A line load is transferred to members that are inside the bounding box of the line load, and whose part names match the part name filtering criteria of the load. The load must have a perpendicular component to the part to be applied to the part. If several members receive the load, the load is distributed based on the length of each member.
Area and uniform loads	Area loads are decomposed to line loads. These decomposed loads are then applied to members. Members inside the bounding box of the load and whose names match the part name criteria receive the load. The area load is divided among the members so that the load applied to the member is proportional to the projection length of the member to the load plane. The resultant of the line loads is the same as the resultant of the original area load.
Nodal load	<p>Tekla Structures binds loads to nodes or members in the analysis model. A load is a nodal load if:</p> <ul style="list-style-type: none"> • It is between two nodes and the distance to the nearest node is less than 110 mm. • It is not between two nodes (even outside the member) but inside the bounding box and meets the part name filtering criteria. <p>Nodal loads do not cause parts to bend.</p>
Member load	If a load does not meet the criteria for the nodal load, it is a member load. Member loads cause parts to bend.
Other loads	Temperature loads are like line loads which affect an entire member. The left, right, top, and bottom surfaces of the member a temperature load affects define the direction of the load.

Load modeling code

Use the **Analysis load modeling** dialog box to determine the building code and safety factors Tekla Structures uses in load combination.

1. Click **Setup > Analysis load modeling...** to open the dialog box.
2. Select the code in the **Load modeling code** list box.
3. Change load combination factors on the appropriate tab if needed:

Tab	Description	More information
Code	The code to follow in analysis and load combination.	

Tab	Description	More information
Eurocode	The partial safety factors in limit states and reduction factors, for the Eurocode, based on load group types.	Load combination factors (p. 50)
British	The partial safety factors in limit states, for the British code, based on load group types.	
AISC (US)	The partial safety factors in limit states, for the US code, based on load group types.	
CM66 (F)	The partial safety factors in limit states, for the French code for steel structures, based on load group types.	
BAEL91 (F)	The partial safety factors in limit states, for the French code for concrete structures, based on load group types.	

4. Click **OK**.



If you have to change the code during a project, you will also need to change the load group types and check load combinations.

Analysis method

You can use either the linear (first order), or non-linear (second order, P-delta), analysis method in Tekla Structures. The non-linear method considers the non-linear nature of the geometry. This takes into account major deflections, but not the non-linear nature of materials. Tekla Structures treats materials as linear. See also [Analysis method](#) (p. 46).

2

Loads

Introduction

Once you have modeled physical structures by creating parts you can start adding loads. In Tekla Structures, you can create point loads, line loads, area loads with uniform or variable distribution. You can also model temperature, wind, and seismic loads. Either attach loads to specific parts or to locations.

In this chapter

This chapter explains how to create and group loads. It also includes a general description of load groups, load types, and load properties. The online help contains step-by-step instructions for all load commands.

Assumed background

We assume that you have created a Tekla Structures model and have a basic understanding of modeling.

Contents

This chapter is divided into the following sections:

- [Basics](#) (p. 25)
- [Grouping loads](#) (p. 27)
- [Load types and properties](#) (p. 30)
- [Distributing loads](#) (p. 35)
- [Working with loads](#) (p. 37)
- [Load reference](#) (p. 40)

2.1 Basics

This section presents some Tekla Structures vocabulary and concepts to help you start to model loads. For additional definitions of terms, see the [Glossary](#) in the Modeling Manual.

Load model	<p>A load model is the portion of the Tekla Structures model that includes all loads, together with the load group and building code information related to them. Each load in a load model has to belong to a load group. Each load can only belong to one load group. A load group can contain one or more loads.</p>
Load group	<p>A load group is a set of loads that are treated alike during load combination. Load groups should contain loads caused by the same action and to which you want to refer collectively. Tekla Structures assumes that all loads in a group:</p> <ul style="list-style-type: none"> • Have the same partial safety and other combination factors • Have the same action direction • Occur at the same time and all together <p>See Grouping loads (p. 27) and Load combination (p. 49).</p> <p>You need to create load groups because the same action can cause different types of loads, for example, point loads and area loads. See Load types (p. 32). You can include as many loads as you like in a load group, of any load type.</p>
Working with loads	<p>In Tekla Structures, you can attach each load to a part for modeling purposes. You can also create floating loads that are bound to locations rather than parts. See Attaching loads to parts or locations (p. 35).</p> <p>Use the load's bounding box and part name filter to define which parts carry the load. See Applying loads to parts (p. 36).</p>

Automatic loads and load groups

Self-weight	<p>Tekla Structures automatically calculates the self-weight of structural parts using the density of the material and the dimensions of the part.</p> <p>To automatically include the self-weight of parts in load combinations, select the Include self-weight checkbox when you create load combinations. See Creating load combinations (p. 52).</p>
Wind loads	<p>Use the Wind load generator (28) tool to define the effects of wind on a structure.</p>
Seismic loads	<p>To automatically include seismic loads in the x and y directions in load combinations:</p> <ol style="list-style-type: none"> 1. Define the code to follow in the seismic analysis. 2. Define the load groups to include in the seismic analysis and their factors. <p>For more information, see Seismic analysis (p. 46).</p>
See also	<p>Load combination types (p. 51)</p> <p>Attaching loads to parts or locations (p. 35)</p>

2.2 Grouping loads

Load groups should contain loads caused by the same action and to which you want to refer collectively. Tekla Structures assumes that all loads in a group:

- Have the same partial safety and other combination factors
- Have the same action direction
- Occur at the same time and all together



You must define at least one load group before you start to create loads. To create a load group, click **Properties > Loads > Load groups...** and use the **Load group properties** dialog box.

Load group properties

Click **Properties > Loads > Load groups...** to open the **Load group properties** dialog box. This is where you define the following properties:

- Name** Each load group must have a unique name. Use load group names to define the visibility and selectability of loads. For example, you can select, modify, or hide loads based on their load group. See [Filter](#) in the Modeling Manual.
- Type** The type of a load group is the type of action that causes the loads.

Actions causing loads are building code specific. See [Load modeling code](#) (p. 22). Most building codes use some or all of the following actions and load group types:

- Permanent, dead, and/or prestressing loads
- Live, imposed, traffic, and/or crane loads
- Snow loads
- Wind loads
- Temperature loads
- Accidental and/or earthquake loads
- Imperfection loads



Tekla Structures automatically determines and applies the self-weight of parts. See [Automatic loads and load groups](#) (p. 26).

- Direction** The direction of a load group is the global direction of the action that causes the loads. Individual loads in a load group retain their own magnitudes in the global or local x, y, and z directions. See also [Load magnitude](#) (p. 34). Load group direction affects which loads Tekla Structures combines in load combination.

Example	For example, a dead load is caused by gravity so it has an action direction of z. Some wind loads are caused by wind from the south (an action direction of y) and others by wind from the west (an action direction of x).
Color	Use different colors for different load groups.

Load group compatibility

When Tekla Structures creates load combinations for structural analysis, it follows the building code you select in **Setup > Analysis load modeling....** See [Load modeling code](#) (p. 22) and [Load combination](#) (p. 49).

To accurately combine loads which have the same load group type, you need to identify which load groups:

- Can occur at the same time (are compatible)
- Exclude each other (are incompatible)

To define load group compatibility, click **Properties > Loads > Load groups....** Enter numbers to indicate compatibility.

Compatibility Compatible load groups can act together or separately. They can actually be one single loading, for example, a live loading that needs to be split in parts acting on different spans of a continuous beam. Tekla Structures then includes none, one, several, or all of the compatible load groups in a load combination.

Incompatibility Incompatible load groups always exclude each other. They cannot occur at the same time. For example, a wind loading from the north is incompatible with a wind loading from the south. In load combination Tekla Structures only takes into account one load group in an incompatible grouping at a time.



Tekla Structures automatically applies basic compatibility facts, such as self-weight being compatible with all other loads, or live loads being compatible with wind load.

Tekla Structures does not combine wind or seismic loads in the x direction with those in the y direction.

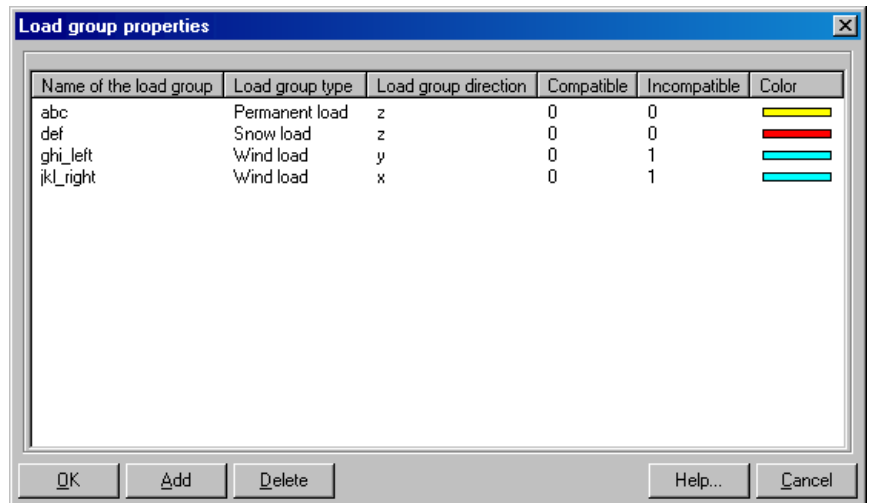
Compatibility indicators are all 0 by default. It indicates that Tekla Structures combines the load groups as defined in the building code.

Working with load groups

You must create at least one load group before you can start to create loads.

Use the **Load group properties** dialog box to view, define, modify, and delete load groups. For example, this is where you set load group properties and indicate load group compatibility.

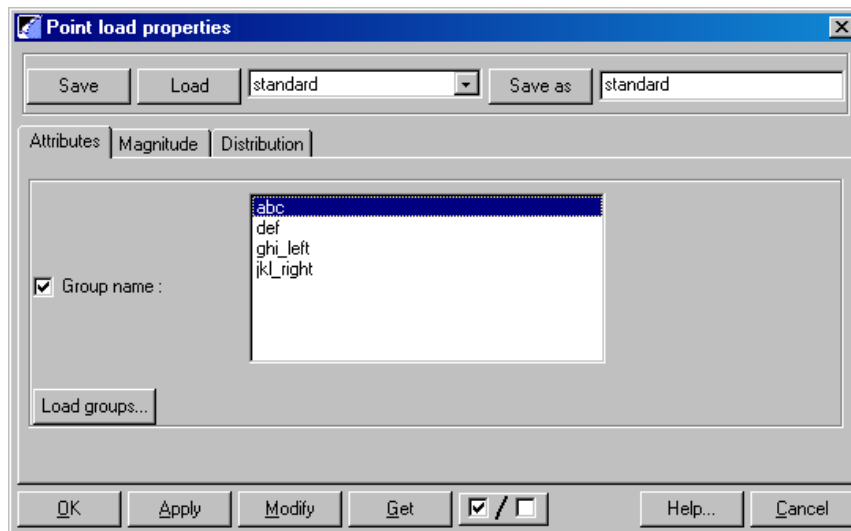
Click **Properties > Loads > Load groups...** to open the dialog box:



Load group types vary according to the code defined in **Setup > Analysis load modeling....** If you have to change the code during a project, you will also need to change the load group types and check load combinations.

Load properties

All existing load groups are also listed on the **Attributes** tab in the load properties dialog boxes. Add the load to a load group here:



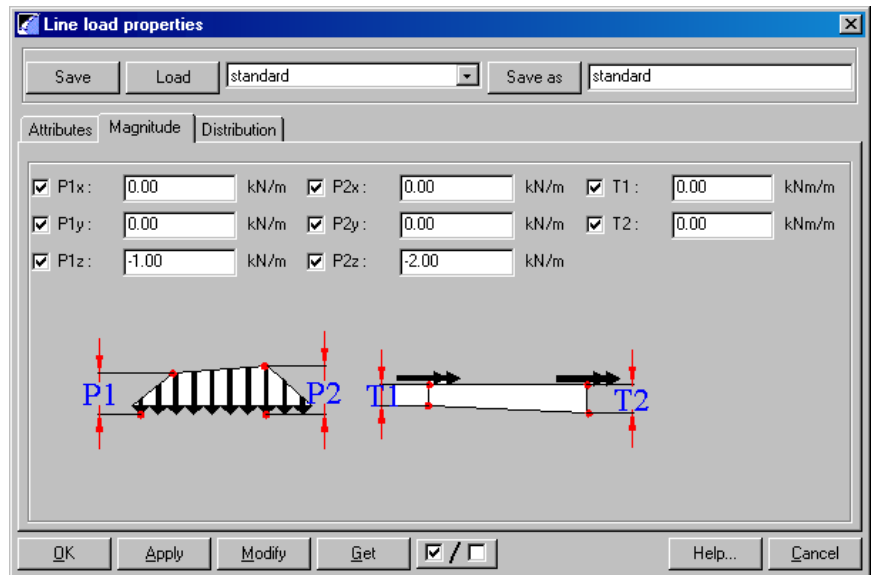
To view the load group properties, click the **Load groups...** button in the load properties dialog box.

2.3 Load types and properties

Introduction

Each load has a type and properties which define it (e.g. magnitude, direction, and distribution). This section describes the different load types and the properties of each load type.

Use the load properties dialog box to view or modify the properties of a load. Click **Properties > Loads** and select a load type to open its properties dialog box.

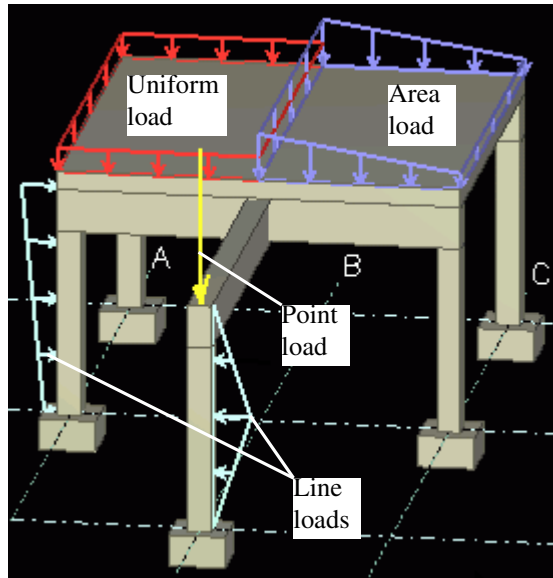


Filtering by properties

You can use load types and groups in filters. For example, you can select, modify, or hide loads based on their type and load group. See [Filter](#) in the Modeling Manual.

Load types

Tekla Structures includes the following load types:



Load type	Description
Point load	A concentrated force or bending moment that can be attached to a part.
Line load	A linearly-distributed force or torsion. By default it runs from one end of the part to the other. You can also create a line load with off-sets from the part ends. A line load can be attached to a part. Its magnitude can vary linearly across the loaded length.
Area load	A linearly-distributed force bounded by a triangle or quadrangle. You do not have to bind the boundary of the area to parts.
Uniform load	A uniformly-distributed force bounded by a polygon. Uniform loads can have openings. You do not have to bind the polygon to parts.

Load type	Description
Temperature load	<ul style="list-style-type: none"> A uniform change in temperature, that is applied to specified parts, and that causes axial elongation in parts. A temperature difference between two surfaces of a part that causes the part to bend.
Strain	An initial axial elongation or shrinkage of a part.







To ensure that load analysis is correct, use area and uniform loads for loads on floors. For example, when the layout of beams changes, Tekla Structures recalculates the loads to the beams. It will not do this if you use point or line loads on individual beams.

Load forms

Distributed loads (line and area loads) can have different load forms.



Line load

The load form of a line load defines how the load magnitude varies along the loaded length. The options are:

Option	Description
	The load magnitude is uniform across the loaded length.
	The load has different magnitudes at the ends of the loaded length. The magnitude changes linearly between the ends.
	The load magnitude changes linearly, from zero at the ends of the loaded length, to a fixed value in the middle of the loaded length.
	The load magnitude changes linearly, from zero at one end of the loaded length, through two (different) values, back to zero at the other end.

Area load

The load form of an area load defines the shape of the loaded area. It can be:

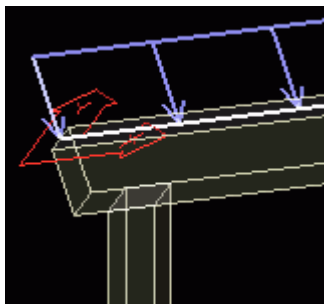
Option	Description
	Quadrangular
	Triangular

For information on how to define the length or area a load affects, see [Loaded length or area](#) (p. 37).

Load magnitude

Load magnitude can occur in x, y, and z directions. The coordinate system is the same as the current work plane, so positive coordinates indicate a positive load direction. See [Basics](#) in the Modeling Manual.

For example, when you create loads perpendicular to sloped parts, shifting the work plane helps you to place loads accurately. See [Defining the work area and shifting the work plane](#) in the Modeling Manual.



Some types of loads can have several magnitude values. For example, the magnitude of line and area loads may vary along the loaded length or across an area. See [Load forms](#) (p. 33).

In the load properties dialog boxes, the following letters denote magnitudes of different types:

- **P** is for a force acting on a position, along a line, or across an area.
- **M** is for bending moments acting on a position or along a line.
- **T** is for torsional moments acting along a line.

Temperature loads and strain

Temperature loads can be caused by:

- An increase or decrease in temperature
- A difference in temperature between the top and bottom surfaces of a part
- A difference in temperature between the sides of a part

Temperature changes cause axial elongation or uniform volume expansion in parts.

Different surface temperatures cause parts to bend.

Strain

Strain is an initial axial elongation (+) or shrinkage (-) of a part.

To define the temperature loads and strain that affect parts, click **Properties > Loads > Temperature load...** and use the **Magnitude** tab.

2.4 Distributing loads

This section explains how to attach loads and how to define which parts, or lengths and areas of parts, carry loads.

Attaching loads to parts or locations

You can attach loads to parts or locations for modeling purposes.

Attaching a load to a part binds the load and the part together in the model. If the part is moved, copied, deleted, etc., it affects the load. For example, a prestressing load moves with the part to which it is attached, and disappears if the part is deleted.

If you do not attach a load to a part, Tekla Structures fixes the load to the position(s) you pick when you create the load.

To attach a load to parts or locations, open the load properties dialog box. On the **Distribution** tab, select an option in the **Load attachment** list box:

Option	Description
Attach to member	Attaches the load to a specific part. If the part is moved, copied, deleted, etc., it affects the load.
Don't attach	The load is not attached but it is considered a floating load. This load is bound to the position you pick when you create the load, not to parts.



If you select the **Attach to member** option, you must select the part before picking the position for the load.

To define which parts carry a load, see [Applying loads to parts](#) (p. 36).

Applying loads to parts

In order to apply loads in the structural analysis model, Tekla Structures searches for parts in the areas that you specify. For each load, you can define the load-bearing parts by name and the search area. To do this, open the load's properties dialog box and click the **Distribution** tab.

Part names

To define the parts that carry the load, enter the part names in the **Part names** field and select **Include** in the list box.

To define the parts that do not carry the load, enter the part names in the **Part names** field and select **Exclude** in the list box.



You can use wildcards when listing the part names. See [Using wildcards](#) in the Modeling Manual.

Bounding box

Use the load's **bounding box** to define the area to search for the parts that carry the load. The bounding box is the volume around the load that Tekla Structures searches for load-bearing parts.

Each load has its own bounding box. You can define the dimensions of a bounding box in the x, y, and z directions of the current work plane. The dimensions are measured from the reference point, line, or area of the load. See also [Handles](#) (p. 38).

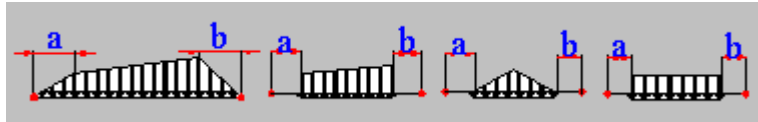
Offset distances from the reference line or area do not affect the size of the bounding box. See [Loaded length or area](#) (p. 37).

Loaded length or area

If a line, area, or uniform load affects a length or area which is difficult to pick in the model, pick one close to it. Then use the values in the **Distances** fields in the load properties dialog boxes to pinpoint the length or area. You can shorten or divide the loaded length, and enlarge or reduce the loaded area.

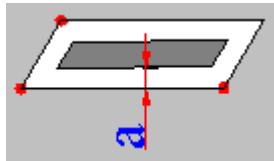
Line load

To shorten or divide the length of a line load, enter positive values for **a** and **b**.



Area load

To enlarge the area an area load affects, enter a positive value for **a**. To reduce the area, enter a negative value.



2.5 Working with loads

To modify the properties of a load, double-click it in the model to open the relevant load properties dialog box.

When you have finished, click **Modify** to update the properties of the load in the model.

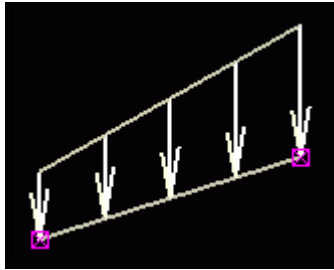
Changing loaded length or area

As well as changing load properties, you can modify loads by:

- Moving line load ends
- Moving uniform load corners
- Changing the shape of openings in uniform loads
- Adding corners to uniform loads

Handles

Tekla Structures indicates the load **reference points** (the ends and corners of line, area, and uniform loads) using **handles**. When you select a load, the handles are magenta.



You can use these handles to move load ends and corners:

1. Select the load to display its handles.
2. Click the handle you want to move. Tekla Structures highlights the handle.
3. Move the handle(s) like any other object. See [Move](#) in the Modeling Manual. If you have **Drag and drop** on, just drag the handle to a new position. See [Drag and drop](#) in the Modeling Manual.

To add corners to uniform loads, use the **Polygon shape** command. See [Edit > Polygon shape](#) in the online help.

Scaling loads in model views

You can have Tekla Structures scale loads when you are modeling. This ensures that loads are not too small to see, or so large that they hide the structure.

To scale loads in model views, click **Setup > Analysis load modeling...** and go to the **Load size** tab:

Analysis load modeling

Save Load Save As

Load size Code Eurocode British AISC (US) CM66 (F) BAEL91 (F)

Point load

Minimum size kN equals mm

Maximum size kN equals mm

Line load

Minimum size kN/m equals mm

Maximum size kN/m equals mm

Area load

Minimum size kN/m² equals mm

Maximum size kN/m² equals mm

Uniform load

Minimum size kN/m² equals mm

Maximum size kN/m² equals mm

Temperature load

Minimum size K equals mm

Maximum size K equals mm







OK Cancel

Example

You define that point loads with magnitude of 1 kN or less are 250 mm high in the model, and point loads with magnitude of 10 kN or more are 2500 mm high. Tekla Structures linearly scales all point loads that have a magnitudes between 1 kN and 10 kN between 250 mm and 2500 mm.

2.6 Load reference

To create loads, use the icons on the **Loads** toolbar or select a command from the **Loads** menu. The following table lists the commands for creating loads and gives a short description of each one. For the detailed instructions, see the online help.

Command	Icon	Description
Load groups...		Displays the Load group properties dialog box.
Point load		Creates a point load at a picked position.
Line load		Creates a line load between two picked points.
Area load		Creates an area load using three picked points.
Uniform load		Creates a uniformly-distributed polygonal area load using at least three picked points.
Temperature load		Defines a temperature change in a part, or a temperature difference between two part surfaces.
Wind load generator (28)		Creates wind loads on a structure.

3

Analysis and Design

Introduction

This chapter explains how to run structural analysis in Tekla Structures. It also includes a general description of analysis and design model properties and an overview of analysis commands.

Audience

This chapter is for engineers who run structural analysis on concrete and steel structures.

Assumed background

We assume that you have read the [Chapter 1, Getting Started with Analysis](#), first and defined the support conditions for parts.

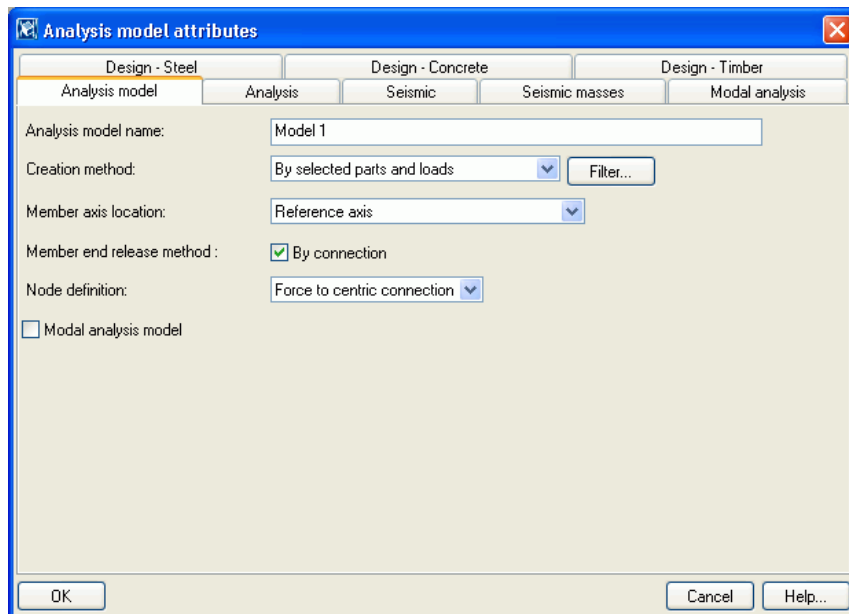
Contents

This chapter is divided into the following sections:

- [Analysis model properties](#) (p. 42)
- [Load combination](#) (p. 49)
- [Working with analysis and design models](#) (p. 56)
- [Structural design](#) (p. 60)
- [Analysis and design reference](#) (p. 61)

3.1 Analysis model properties

This section explains the properties that Tekla Structures uses to create analysis members and analyze them. These properties apply to all members in an analysis model.



Tekla Structures also takes into account the individual analysis properties defined in the parts' properties. See [Determining member properties \(p. 8\)](#).

Creating an analysis model

To set the properties for a new analysis model, click **Analysis > New model...** to open the **Analysis model attributes** dialog box. To create the analysis model, click the **OK** button.

To view or modify the properties of an existing analysis model:

1. Click **Analysis > Analysis & Design models...**
2. Select the model.
3. Click **Properties...**

Model name

Each analysis model must have a unique name, which you can define. For example, you could use a name that describes the portion of the physical model you want to analyze.

Objects in an analysis model

You can define which objects to include in an analysis model. For example, you can create analysis models of:

- The entire physical and load models
- A particular sub structure under a specific loading
- A single part

Creation method

To define which objects to include in an analysis model, open the **Analysis model attributes** dialog box. On the **Analysis model** tab, select an option from the **Creation method** list box. The options are:

Option	Description
Full model	Includes all main parts and loads. Tekla Structures automatically adds physical objects to the analysis model when they are created.
By work area	Includes all the main parts and loads that are inside or partly inside the work area when you create the analysis model.
By selected parts	Only includes selected parts.
By selected parts and loads	Only includes selected parts and loads.
Floor model by selected parts and loads	Only includes selected columns, slabs, floor beams, and loads. Tekla Structures replaces columns in the physical model with supports.

Tekla Structures ignores some objects in the analysis. See [A closer look at the analysis model \(p. 19\)](#).

Analysis model filter

To use the analysis model filter to select objects to include in an analysis model, click the **Filter** button on the **Analysis model** tab in the **Analysis model attributes** dialog box. The analysis model filter works in similar way to the select filter (see [Select filter](#) in the Modeling Manual), but Tekla Structures saves the settings with the analysis model properties. So you can go back and check the criteria you used to select objects.

Tekla Structures automatically adds new objects you create in the physical model to the analysis model if they fulfill the criteria in the analysis model filter.



Use the analysis model filter to filter out non-structural parts, such as railings, from the analysis model.

See also

To add individual objects to an analysis model, or to remove them, see [Adding or removing analysis objects](#) (p. 57).

To check which objects are included in an analysis model, see [Checking objects contained in an analysis model](#) (p. 58).

Member axis

The locations of the member axes of parts define where the analysis members actually meet, and their length in the analysis model. They also affect where Tekla Structures creates nodes.

To define member axis locations for all members in an analysis model, open the **Analysis model attributes** dialog box. On the **Analysis model** tab, select an option from the **Member axis location** list box. The options are:

Option	Description
Neutral axis	The neutral axis is the member axis for all parts. The location of the member axis changes if the profile of the part changes.
Reference axis	The part reference line is the member axis for all parts. See also Part location in the Modeling Manual.
Reference axis (eccentricity by neutral axis)	The part reference line is the member axis for all parts. The location of the neutral axis defines axis eccentricity.
Model default	The member axis of each part is defined individually according to the part's properties. See Member axis location (p. 10).



If you select the **Neutral axis** option, Tekla Structures takes the part location and end offsets into account when it creates nodes. See [End offsets](#) in the Modeling Manual. If you select either of the **Reference axis** options, Tekla Structures creates nodes at part reference points.

Member end connectivity

You can choose to define the support conditions of individual analysis members according to the part's properties, or the support conditions of connections between parts. See [Support conditions](#) (p. 14).

To have Tekla Structures use the support conditions of connections:

1. For each connection or detail, open the properties dialog box. On the **Analysis** tab, select **Yes** in the **Use analysis restraints** list box. See [Analysis properties of components \(p. 13\)](#).
2. Open the **Analysis model attributes** dialog box for an analysis model. On the **Analysis model** tab, select the **By connection** checkbox against **Member end release method**.

Leave the **By connection** checkbox blank to have Tekla Structures use the support conditions of individual parts.

Defining nodes

If physical parts collide but their member axes do not intersect, Tekla Structures is unable to create a common node for them in the analysis model. In order to force the members to meet in the analysis model, Tekla Structures may need to use the methods described in [A closer look at the analysis model \(p. 19\)](#).

Node definition method

If Tekla Structures cannot merge nodes because they are outside the merge distance (**XS_AD_NODE_COLLISION_CHECK_DISTANCE**), you can define how to connect members. Open the **Analysis model attributes** dialog box. On the **Analysis model** tab, select one of the following options in the **Node definition method** list box:

Option	Description
Use rigid links	Creates a node at each member's axis and connects the members using a rigid link between the nodes.
Force to centric connection	Creates a single node for the members and forces them into a centric connection by extending the member axes.

Rigid links

Rigid links have the following properties in the analysis model:

- Profile = PL300.0*300.0
- Material = RigidlinkMaterial
- Density = 0.0
- Modulus of elasticity = $100 \cdot 10^9$ N/m²
- Poisson's ratio = 0.30
- Thermal dilatation coefficient = 0.0 1/K

Analysis method

To define the analysis method for the model, open the **Analysis model attributes** dialog box. On the **Analysis** tab, select an option from the **Analysis method** list box. The options are:

Option	Description	More information
1st order	Linear analysis method.	
2nd order	Geometrically non-linear analysis method.	
P-delta	A simplified second order analysis method. This method gives accurate results when deflections are small.	P-Delta Analysis in the online help

If you select **2nd order** or **P-delta**, Tekla Structures takes into account the additional stresses induced by the deflections of the structure. This leads to the iteration of deflection.

Iteration

The accuracy of the second order analysis depends on the number of iterations; the longer the iteration goes on, the more accurate the analysis. Increasing the number of iterations also increases processing time and analysis model size. To limit the number of iterations in second order analysis, enter a value in the **Maximum number of iterations** field. You can also set the accuracy, which is the relative tolerance used to control the iteration of deflection.

Iteration stops when the analysis reaches the accuracy or the maximum number of iterations you define on the **Analysis** tab.

Seismic analysis

Type

To define which building code to use to generate seismic loads, use the **Seismic** tab in the **Analysis model attributes** dialog box. You can create lateral seismic loads in the x and y directions according to several codes using a static equivalent approach (z is the direction of the gravity loads). Select one of the following options in the **Type** list box:

Option	Description	More information
None	Seismic analysis not run.	
UBC 1997	Uniform Building Code 1997	UBC 1997 Load Definition in the online help
UBC 1994	Uniform Building Code 1994	UBC 1994 or 1985 Load Definition in the online help

Option	Description	More information
IBC 2000	International Building Code 2000	IBC 2000 Load Definition in the online help
IS 1893-2002	Indian Standard. Criteria for Earthquake Resistant Design of Structures	Definition of Lateral Seismic Load per IS:1893 in the online help

Properties

Depending on the code you select, you can define some or all of the following:

- Whether to calculate the accidental torsion
- Seismic zone coefficient (**Zone**)
- Importance factor
- Numerical coefficients R_w for the lateral loads in x and y directions
- Soil profile type
- Soil factor
- Site class
- SDS, SD1, S1
- Near source factors NA and NV
- CT value to calculate time period
- Periods of structure (in seconds) in the x and y directions
- Response reduction factor
- Type of structure
- Damping ratio
- Depth of foundation below ground level

Seismic loads

Use the **Seismic masses** tab in the **Analysis model attributes** dialog box to define the load groups and load group factors to include in the seismic analysis.

To include the self-weight of parts in the seismic analysis, select the **Include self-weight as seismic mass** checkbox.

To move load groups between the **Selected load groups** and **Not selected load groups** lists, select a load group and use the **Add** and **Remove** buttons.

Design codes and methods

Use the **Design** tabs in the **Analysis model attributes** dialog box to define the code and method to use in structural design. Tekla Structures includes several design codes for steel, concrete, and timber. The design options available vary depending on the material.

Steel

The design code options for steel are:

Code	Description
EC3	Eurocode 3
BS	British Standard 5950
AISC ASD	Allowable Stress Design Specification by American Institute of Steel Construction
AISC LRFD	Load and Resistance Factor Design Specification by American Institute of Steel Construction
CM66	French code
AISI	American code
IS800	Indian code IS 800
CSA-S16	Canadian code
EA95	Spanish code
NEN6770	Dutch code
DIN18800	German code DIN 18800

The design method options for steel are:

Method	Description
None	Tekla Structures only runs the structural analysis and creates data on stresses, forces, and displacements.
Check design	Tekla Structures checks whether the structures fulfill the criteria in the design code (i.e. whether cross sections are adequate).

Concrete

The design code options for concrete are:

Code	Description
EC2	Eurocode 2
ACI	American Concrete Institutes's publication 318, Building Code Requirements for Structural Concrete
BS8110	British Standard 8110
BAEL	French code
IS456	Indian code IS 456

Code	Description
CSA-A23	Canadian code
EH91	Spanish code
NEN6720	Dutch code
DIN1045	German code DIN 1045

The design method options for concrete are:

Method	Description
None	Tekla Structures only runs the structural analysis and creates data on stresses, forces, and displacements.
Calculate required area	Tekla Structures defines the required area of reinforcement.

Timber

There are currently no design options available for timber.

Design properties

When you select a design code and method for a material, Tekla Structures lists the design properties in the lower part of the **Design** tab in the **Analysis model attributes** dialog box. Click on an entry in the **Value** column to change the value of a particular property.

To change the design properties of specific parts, use the **Design** tab in the appropriate part properties dialog box. See [Design information \(p. 17\)](#).

3.2 Load combination

Introduction

Load combination is a process in which some simultaneously acting load groups are multiplied by their partial safety factors and combined with each other according to specific rules.

Load combination rules are specific to a design process and are defined in building codes. One of the most typical design processes is the limit state design.

The result of the load combination process is a load combination.

You can have Tekla Structures automatically create load combinations, or you can create and modify them manually.

Load combination properties

Load combination properties define how Tekla Structures combines loads. The following properties control the load combination process:

- [Load modeling code](#) (p. 22)
- [Load combination factors](#) (p. 50)
- [Load combination types](#) (p. 51)
- [Load group compatibility](#) (p. 28)

Name and ID

Each load combination must have a unique name. Use names that describe the load situation.

Each load combination has an ID. This is an incremental number, based on order in which load combinations are created in the analysis model.

Load combination factors

You can use values for load combination factors that are building-code specific or user-defined. To use building-code specific factors, click **Setup > Analysis load modeling...** On the **Code** tab, select an option from the **Load modeling code** list box. See [Load modeling code](#) (p. 22).

If you change any values on the code-specific tabs, save the properties using a new name. To do this, enter a name in the field next to the **Save as** button and click the **Save as** button.



You should not need to change these settings during the project. If you have to, you will also need to change the load group types and check load combinations.

Partial safety factors

The partial safety factors needed in the limit state design appear on the code-specific tabs. They are:

- Unfavorable partial safety factor in the ultimate limit state (γ_{sup})
- Favorable partial safety factor in the ultimate limit state (γ_{inf})
- Unfavorable partial safety factor in the serviceability limit state (γ_{sup})
- Favorable partial safety factor in the serviceability limit state (γ_{inf})

Reduction factors

Depending on the codes you use, you may need to use other combination factors. For example, the Eurocode contains three reduction factors (ψ_0 , ψ_1 , ψ_2). Reduction factors exclude the impractical effects of simultaneous loads.

Load combination types

You must create an analysis model before you can define load combination types. See [Creating an analysis model](#) (p. 42).

You can perform several types of load combination, which vary according to the building code you use. The options are:

Combination type	Description	Applies to
Load groups (LG)	Each load group forms a load combination. All partial safety factors equal 1.00.	All codes
Ultimate limit state (ULS)	Combines load groups that occur persistently and transiently. Uses the partial safety factors of the ultimate limit state when combining loads.	Eurocode, British, AISC
Serviceability limit state – Rare (SLS RC)	Combines load groups that occur quasi-permanently and rarely. Uses the partial safety factors of the serviceability limit state when combining loads.	Eurocode
Serviceability limit state – Quasi-permanent (SLS QP)	Combines load groups that occur quasi-permanently. Uses the partial safety factors of the serviceability limit state when combining loads.	Eurocode
Serviceability limit state (SLS)	Combines load groups that occur quasi-permanently. Uses the partial safety factors of the serviceability limit state when combining loads.	ASCE
Normal loads	Combines load groups and uses factors according to the French codes CM66 or BAEL91.	CM66, BAEL91
Extreme loads		CM66
Displacement loads		CM66
Accidental loads		CM66
Ultimate loads		BAEL91
Ultimate accidental loads		BAEL91

Use the **Load combination generation** dialog box to define the load combination type.

Automatically including loads in combinations

You can automatically include various loads in load combinations. To do this, select the appropriate checkboxes in the **Load combination generation** dialog box. The options are:

Checkbox	Description	Applies to
Include self-weight	Automatically includes the self-weight of parts in load combinations. This means that you do not have to model self-weight loads separately. See Automatic loads and load groups (p. 26).	All codes
Generate wind also in opposite direction	If the analysis model has wind loads from a specific direction (x or y), select this checkbox to include wind loads from the opposite direction (-x or -y).	All codes



To automatically include seismic loads in load combinations, use the **Seismic** and **Seismic masses** tabs in the **Analysis model attributes** dialog box. See also [Seismic analysis](#) (p. 46).



If the analysis model has imperfection loads, Tekla Structures automatically creates load combinations with both the positive and negative directions (x and -x, or y and -y).

Creating load combinations

To create load combinations:

1. Click **Analysis > Analysis & Design models...**
2. In the **Analysis & Design models** dialog box, select an analysis model and click **Load combinations...** to open the **Load combinations** dialog box. This lists the existing load combinations, together with their ID, name, type, and the load groups they contain.

Load combinations

Save Load Save as

Id	Name of the combination	Combination type	Self weight	Permanent load	Live load	Wind load in x-direction
8	ULS8	ULS	1.00x1.00	1.00x1.00		1.00x1.50
9	ULS9	ULS	1.00x1.35	1.00x1.35	1.00x1.50	0.60x1.50
10	ULS10	ULS	1.00x1.35	1.00x1.35	0.70x1.50	0.60x1.50
11	ULS11	ULS	1.00x1.00	1.00x1.00	1.00x1.50	0.60x1.50
12	ULS12	ULS	1.00x1.00	1.00x1.00	0.70x1.50	0.60x1.50
13	ULS13	ULS	1.00x1.35	1.00x1.35	0.70x1.50	1.00x1.50
14	ULS14	ULS	1.00x1.00	1.00x1.00	0.70x1.50	1.00x1.50
15	SLSQP15	SLS_QP	1.00x1.00	1.00x1.00		
16	SLSQP16	SLS_QP	1.00x1.00	1.00x1.00	0.30x1.00	

New... Generate... Number of combinations: 16 Remove Remove all

OK Apply Cancel

Use the buttons in the **Load combinations** dialog box to carry out various tasks. The buttons are:

Button	Description
New...	Displays the Load combination coefficients dialog box, where you manually create load combinations. See Manual load combination (p. 55).
Generate...	Automatically generates load combinations based on the code and factors in Setup > Analysis load modeling.... See Automatic load combination (p. 54).
Remove	Deletes the selected load combination.
Remove all	Deletes all load combinations.

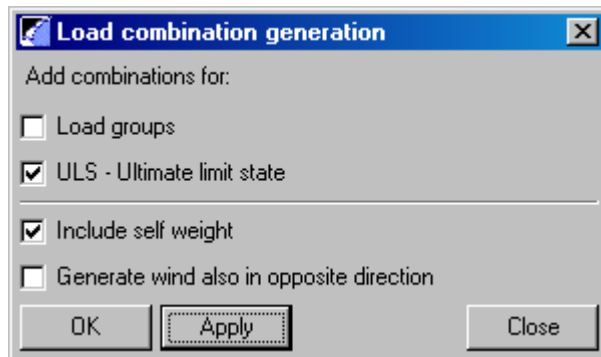


Use the **Save as** and **Load** buttons to copy load combinations between analysis models.

Automatic load combination

To automatically create load combinations:

1. In the **Load combinations** dialog box, click **Generate...** to open the **Load combination generation** dialog box.



2. In the upper part of the dialog box, select the checkboxes against the combinations you want to create. See [Load combination types \(p. 51\)](#).
3. To automatically include the self-weight of parts or wind loads from the opposite direction, select the appropriate checkboxes in the lower part of the dialog box. See [Automatically including loads in combinations \(p. 52\)](#).
4. Click **Apply** or **OK**.

Tekla Structures creates the load combinations for different load groups and limit states according to the load modeling code you select, and uses the combination factors defined in **Setup > Analysis load modeling....**

See also [Creating load combinations \(p. 52\)](#) and [Manual load combination \(p. 55\)](#).

Manual load combination

To create load combinations manually:

1. In the **Load combinations** dialog box, click **New...** to open the **Load combination coefficients** dialog box.

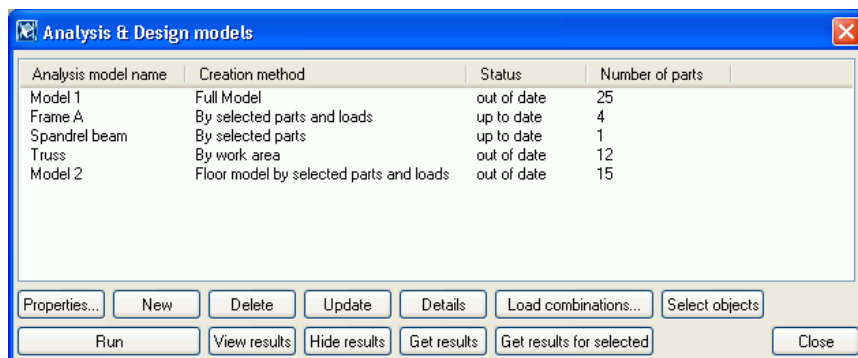
Sign	Reduction_Factor	Partial Safety Factor	Load Group
+	1.00	1.20	Self Weight
+	1.00	1.60	snow
+	0.50	1.60	wind right
+	1.00	1.60	live1

2. Select a combination type. See [Load combination types \(p. 51\)](#).
3. Enter a unique name for the load combination. Try to make the name as descriptive as possible.
4. Use the sign and arrow buttons to move load groups between the **Loads available** list and the **Combinations** table.
5. Modify the combination factors in the **Combinations** table by clicking a value.
6. Click **Apply** or **OK**.

See also [Creating load combinations \(p. 52\)](#) and [Automatic load combination \(p. 54\)](#).

3.3 Working with analysis and design models

This section explains how to examine and modify analysis models. Click **Analysis > Analysis & Design models** and use the **Analysis & Design models** dialog box.



See also [Creating load combinations \(p. 52\)](#)

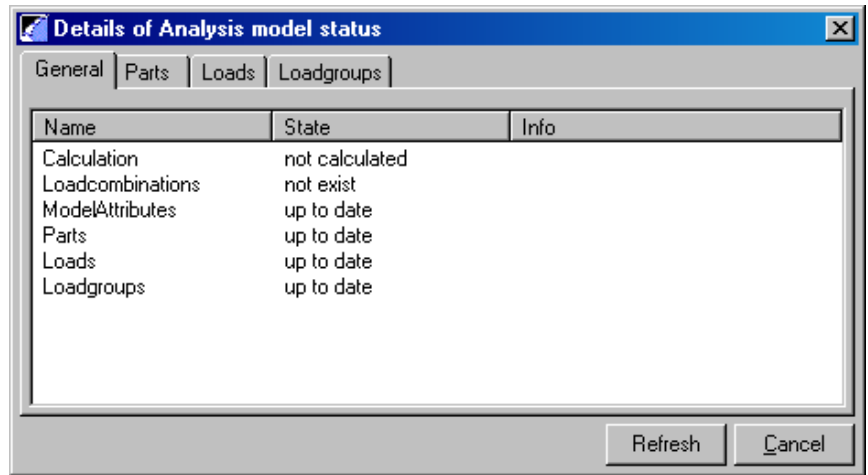
Analysis model status

The **Status** field in the **Analysis & Design models** dialog box shows the status of the analysis model. This can be:

- **up to date**
- **out of date**

Out of date means that the model has changed. To find out more:

1. Click **Analysis > Analysis & Design models...**
2. In the **Analysis & Design models** dialog box, select an analysis model.
3. Click **Details** to open the **Details of analysis model status** dialog box. This displays the status of the following:
 - Calculation
 - Load combinations
 - Analysis model properties
 - Parts
 - Loads
 - Load groups



The **Parts**, **Loads**, and **Load groups** tabs display information about individual objects based on their ID number.

Adding or removing analysis objects

As well as changing the properties of an analysis model, you can also modify existing analysis models by adding and removing objects.



Adding and removing objects changes the analysis model status to **out of date**, so you must run the analysis again. See also [Analysis model status \(p. 56\)](#).

To add or remove parts and loads:

1. Click **Analysis > Analysis & Design models....**
2. In the **Analysis & Design models** dialog box, select a model.
3. In the physical model, select the parts and loads to add or remove.
4. To add the objects to the analysis model, click **Analysis > Add members**.
5. To remove the objects from the analysis model, click **Analysis > Remove members**.

Checking objects contained in an analysis model

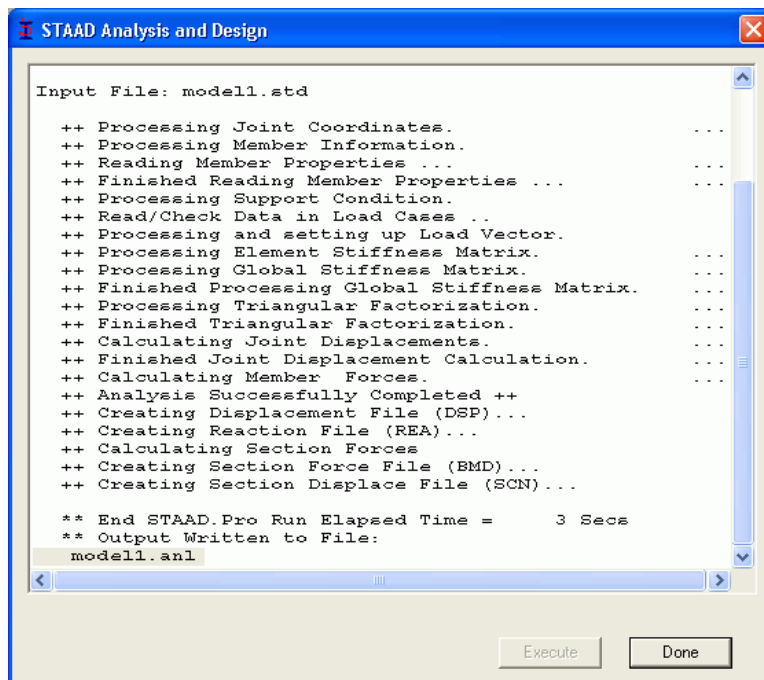
To check which parts and loads an analysis model contains:

1. Click **Analysis > Analysis & Design models....**
2. In the **Analysis & Design models** dialog box, select a model.
3. Click the **Select members** button. Tekla Structures highlights and selects the parts and loads in the physical model.

Running analysis

To run structural analysis on an analysis model:

1. Click **Analysis > Analysis & Design models....**
2. In the **Analysis & Design models** dialog box, select a model.
3. Click the **Run** button. The analysis engine starts and opens the **STAAD Analysis and Design** window, which displays information about each step in the analysis.



4. If the analysis is not complete, you will see the following message in the **STAAD Analysis and Design** window:

ERROR in Analysis. Check Output (.ANL) File.

The anl file is located in the ..\TeklaStructuresModels*your_model*\Analysis*your_analysis_model* folder.
5. Click the **Done** button.
6. To view the results, go to the **Analysis & Design models** dialog box and click the **View results** button.

Viewing analysis results

Once you have run the analysis, you can view the results. For information on analysis results, see the following topics in the online help:

- [Introduction](#) to Post Processing
- [Analysis > Results](#)
- [Analysis > Member results](#)
- [Analysis > Get results](#)
- [Analysis > Analysis & Design models...](#)

So that you can perform a visual check, you can have Tekla Structures use different colors to show the utilization ratio of steel parts in the physical model. To do this:

1. Run the analysis.
2. Click **Analysis > Analysis & Design models....**
3. In the **Analysis & Design models** dialog box, select an analysis model.
4. In the Model Editor, click **Setup > Colors....**
5. In the **Colors** dialog box, select **Analysis utilization check** from the **Color by** list box.
6. Set the ranges of ratio for each of the colors that Tekla Structures uses to show safe and unsafe parts.
7. Click **Apply** or **OK**. Tekla Structures shows the utilization ratio of the steel parts in the selected analysis model using the following colors:



To show the utilization ratio of steel parts in a report, add the AD\$\$UtilityRatio template field in the report template you use.

Freezing analysis results

You can freeze analysis results to prevent them being accidentally changed. Freeze analysis results when you want to keep them even if the model changes.

To freeze analysis results:

1. Run the analysis.
2. Click **Analysis > Analysis & Design models....**
3. In the **Analysis & Design models** dialog box, select an analysis model.
4. Click **Analysis > Freeze results.**

If the analysis results are frozen and you try to run the analysis, Tekla Structures will ask if you want to keep the existing results.

Unfreezing analysis results

To unfreeze analysis results, select an analysis model and click **Analysis > Unfreeze results.**

3.4 Structural design

Once the analysis phase is complete, you can move on to design the structure based on the stresses, required reinforcement area, cross section, or profile size in the analysis results.

Optimizing part size

You can have Tekla Structures check the profiles of the steel parts in the physical model and suggest the best profiles to use, based on the analysis results.

Design group

Tekla Structures creates a **design group** of steel parts that have the same name and profile. Tekla Structures uses design groups when it searches for the optimal profiles for parts. It assigns the same profile to all parts in a design group.

Tekla Structures uses the following process to optimize part sizes for each design group:

1. Finds the governing part of the design group, using the results of the analysis.
2. Selects an appropriate profile for the governing part.
3. Checks the entire physical model for steel parts that belong to the design group.
4. Applies the new profile of the governing part to all parts in the design group.



To optimize part sizes, run the analysis and then click **Analysis > Optimize....** For more information, see [Analysis > Optimize](#) in the online help.




Checking design groups

To check which parts a design group contains, select the design group in the **Optimization results** dialog box. Tekla Structures highlights and selects the corresponding parts in the model.

3.5 Analysis and design reference

Use the commands on the **Analysis** menu to work with analysis and design models. The following table lists the analysis and design commands and gives a short description of each one. For the detailed instructions, see the online help.

Command	Icon	Description
New model...		Displays the Analysis model attributes dialog box and creates a new analysis model.
Analysis & Design models...		Displays the Analysis & Design models dialog box so that you can work with analysis and design models.
Load combinations...		Displays the Load combinations dialog box so that you can work with load combinations.

Command	Icon	Description
Select members		Highlights and selects the parts and loads that are included in the analysis model in the physical model. See also Checking objects contained in an analysis model (p. 58) .
Add members		Adds parts and loads to the selected analysis model.
Remove members		Removes parts and loads from the selected analysis model.
Run		Runs the analysis on the selected analysis model. See also Running analysis (p. 58) .
Optimize		Uses the analysis results to optimize the size of steel parts.
Results		Displays the results for the selected analysis model.
Results for selected		Same as Results but only displays the results of the selected parts.
Member results		Displays the analysis results for a selected part.
Hide results		Hides the analysis results and closes the STAAD.Pro postprocessor.
Get results		Saves the maximum axial force, shear force, and bending moment at the part ends as user-defined attributes in the part properties. To view these results, open a part's user-defined attributes dialog box.
Get results for selected		Same as Get results but only saves results for the selected parts.
Freeze results		Freezes or unfreezes the results of a selected analysis model. See Freezing analysis results (p. 60) .
Unfreeze results		

Index

A

analysis and design	
overview	41
prior to	7
analysis engine	6, 18
analysis member offsets.....	11
analysis members	
properties	8, 17
analysis method	23, 46
analysis models	6
a closer look	19
adding or removing objects	57
checking objects.....	58
creating	42
filtering objects	43
freezing results	60
modifying.....	42, 57
objects	43
properties	42
running analysis	58
seismic loads	46
status.....	56
viewing results	59
analysis settings.....	18
applying loads to parts	36
attaching	
loads to parts	35
automatic loads.....	26
in load combinations.....	52
seismic loads.....	47
self-weight	26

B

bounding box.....	36
-------------------	----

C

color by analysis type	9
color by analysis utilization check	59
combination factors	50
combining loads	49
compatibility of load groups	28
components	
in analysis	13
creating	
analysis models	42
load combinations	52
loads.....	40

D

defining	
load groups	28
nodes.....	19, 45
support conditions	14
degree of freedom.....	14
deleting	
load groups	28
design codes and methods	47
design information.....	17
distributing loads.....	35
DOF	
see degree of freedom	14

E

elements.....	8
---------------	---

F

FEM	6
-----------	---

filter	
in analysis models.....	43
filtering	
analysis model objects	43
finite element method	6
freezing	
analysis results	60

H

handles	
of loads	38

I

intermediate analysis members.....	17
iteration	46

L

load combination	49
automatic.....	54
code	22
creating combinations	52
factors.....	22, 50
manual	55
properties	50
types.....	51
load forms.....	33
load groups	26, 27
automatic.....	26
compatibility	28
defining	28
deleting	28
modifying	28
properties	27
load model	6, 26
load modeling code	22
load types	32

loads.....	25, 40
applying.....	36
attaching	35
automatic	26
bounding box	36
combining	49
creating	40
distribution.....	35
forms	33
grouping	27
in analysis	21
magnitude	34
modifying	37
properties.....	30
scaling in model views.....	38
seismic	46
types.....	32
longitudinal member offset	11

M

member analysis type	9
member axis	
of all parts in analysis model	44
of individual parts	10
member end connectivity	44
member end release method	44
members	8
properties.....	8, 17
modifying	
analysis models	42, 57
load groups	28
loads.....	37

N

nodes.....	8
defining.....	19, 45

O

optimizing part size.....	61
---------------------------	----

P

part properties	
optimizing part size.....	61

partial safety factors	50
physical model	6
plates	
in analysis	11

R

reduction factors	50
running analysis	58

S

safety factors	50
seismic analysis	46
seismic loads	46
self-weight	26
strain	35
support conditions	14
defining	14

T

temperature load	35
------------------------	----

U

utilization ratio	59
-------------------------	----